TD-00-045
July 26, 2000

Fermilab

Fermi National Accelerator Laboratory
P.O. Box 500 • Batavia, Illinois • 60510

HFM Cos(θ) Mechanical Model - 1: Fabrication and Test Results

Deepak Chichili, Tug Arkan and Nicolai Andreev
Engineering and Fabrication Department
Technical Division
Fermilab, Batavia, IL 60510

Introduction

A 10 inch long mechanical model was conceived with the following goals:

- To test the magnet assembly procedure
- To shake down the assembly tooling
- To verify the instrumentation plan
- To verify the mechanical analysis

The length of the model was chosen to be 10 inches because it was the maximum length we could EDM the components at one pass. Cable made out of ITER strand was used partly due to availability and partly because we are only interested in the mechanical aspects and not electrical and ITER strands were much cheaper than rest of them. Finally the tooling design for the model was developed at the time when 44.5 mm bore design was considered for the dipole model - 1. This was later changed to 43.5 mm due to increase in insulation thickness from 125 μm to 250 μm. However the mechanical model was left unchanged.

This report has two sections, fabrication and test results. The first section illustrates the fabrication procedure used to build this model, which includes making curved stacks, reaction procedure, epoxy impregnation, instrumentation and finally assembly of the model. The second section describes the test results obtained using capacitance and resistive gauges at various stages of the magnet assembly and also the comparison of this data with ANSYS model.

Fabrication of the Model

Curved Stacks

To reduce the time and cost, we decided to fabricate curved stacks of each quadrant instead of winding a whole half coil and then using the straight section. Cables were cut slightly longer than 10 inches and were insulated with butt-lap ceramic insulation. Ceramic Binder (CTD 1002 x) was then applied to these cables. The wet insulated cables were placed into the slot of the curved stack fixture as shown in the Fig. 1a. The top plate was closed using blots when gives the right geometry for the curved stack. The ends of the cables were then cut using a saw (see Fig. 1b). The fixture was later placed in a furnace at 150 ^{o}C for 30 min to cure the sample. Fig. 1c shows an inner and an outer curved stack.

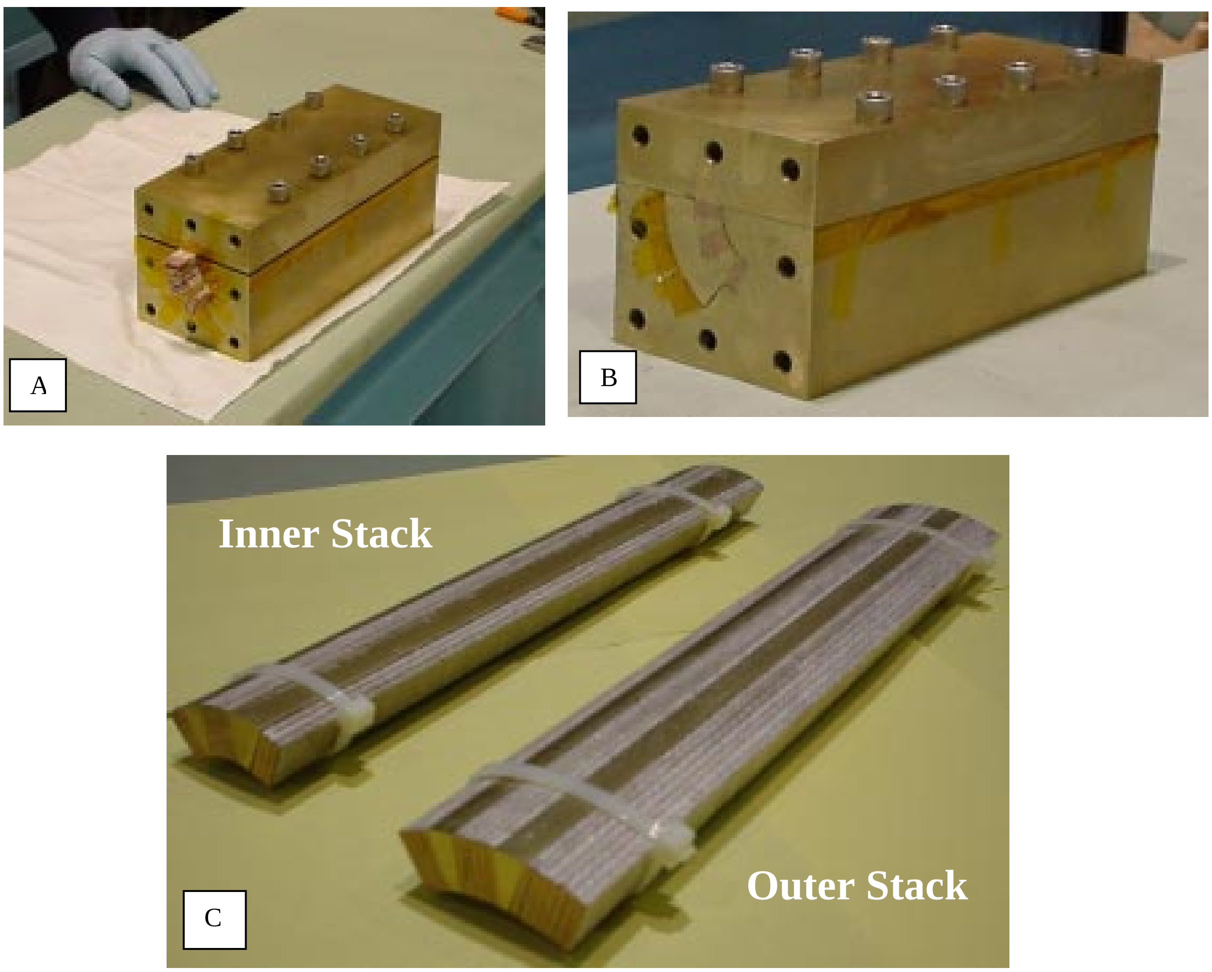
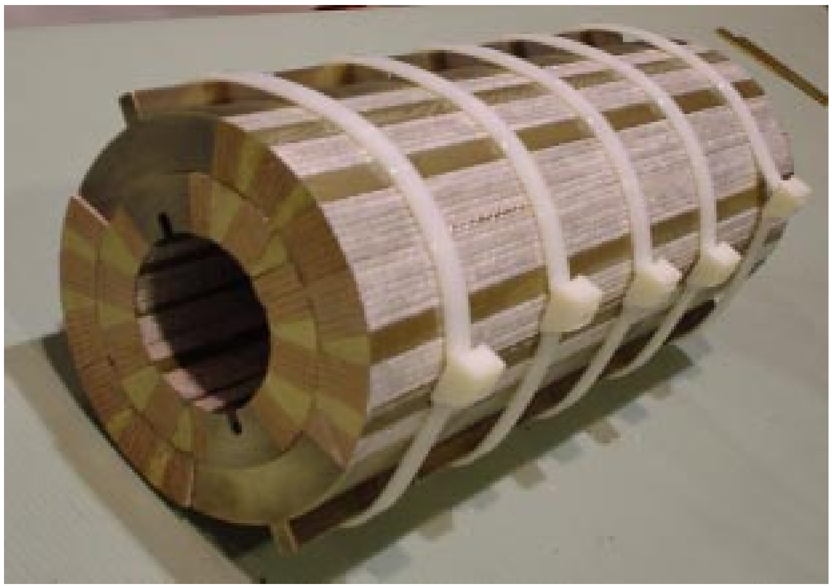


Figure 1: *Fabrication of curved stacks. (a) wet insulated cables placed in the slot of the fixture, (b) after closing the top plate and cutting the ends of the cable and (c) inner and outer curved stacks after curing.*

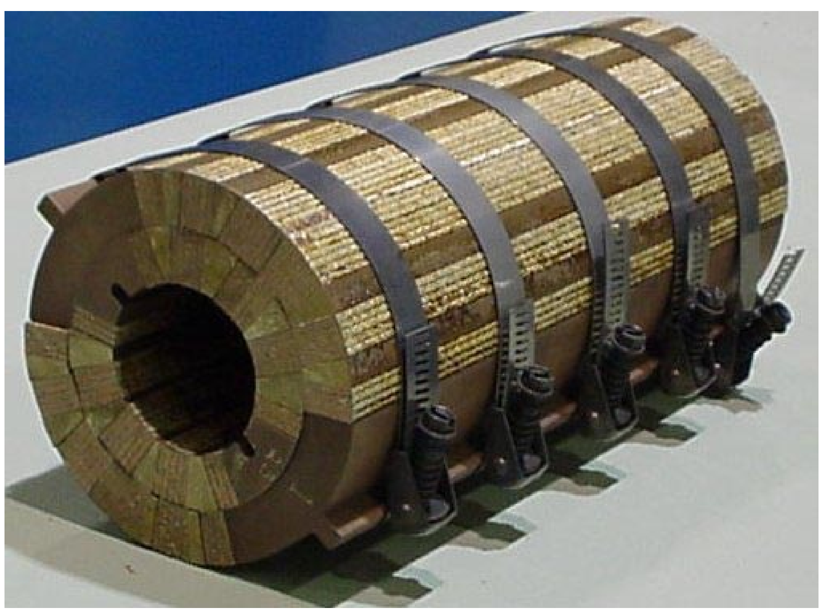
The initial set of curved stacks fell apart once they were taken out of the fixture. We later realized that the cable we received had a residual twist in excess of 135 degrees. The rest of the curved stacks were made with the cable which was heat-treated to 200 °C for 30 min. Note that this initial heat-treatment of the cable reduced the twist from 135 to 40 degrees.

Assembly and Reaction:

Four inner and four outer curved stacks were fabricated and assembled for reaction. Fig. 2 shows the model before and after reaction.



(a)



(b)

Figure 2: *Mechanical model (a) before and (b) after reaction.*

The reaction procedure followed was a two step process as given below:

Step 1: 575 ^{o}C for 200 hr. with a ramp rate of 0.42 ^{o}C/hr
Step 2: 700 ^{o}C for 40 hr. with a ramp rate of 0.44 ^{o}C/hr

After the reaction process, we observed that the cables shrunk with respect to the wedges and the pole pieces which were made out of aluminum bronze. The cable shrinkage is about 2.15 mm/m. Note that it is a commonly observed phenomenon that during reaction process, the thickness of the Nb_3Sn cable increases while the length and the width of the cable decreases.

<u>Epoxy Impregnation</u>:

The reacted model was then epoxy impregnated using a short impregnation tooling. The set up and procedure were similar to that of ten-stack sample impregnation. I would like to point out at this stage that we were not able to close the mold cavity of the impregnation tooling with the reacted model inside. So we had to remove a cable from the mid-plane of the inner coil. This made the inner layer skewed to one side with respect to the outer layer. It was unfortunate that we had to do this, however we went ahead with the model so that we could atleast test the magnet assembly procedure and shake down the tooling. As far the instrumentation and the comparison with the analysis, we should not be very far from reality.

We mold released an inner and two outer pole pieces so that we could put capacitance gauges between them and the coil after impregnation to measure azimuthal stress. Fig. 3 shows the impregnated model along with spacers.

Figure 3: *Impregnated model with instrumentation and aluminum spacers.*

Instrumentation:

Both capacitance and resistive gauges were used to obtain the stress distribution in various components of the model. Capacitance gauges similar to that used in HGQ program were used to measure the azimuthal stress in the coils. They were placed between the pole pieces and the coil in both inner and outer layers. Mid-plane stresses were not measured in the model, but we hope to do so in the next one. The cap gauges were fabricated and calibrated in house. The calibration was done both at room temperature and at 77 K. Figs 4 and 5 show the calibration plots for one of the capacitance gauges at 300 K and at 77 K respectively . Note that the capacitance versus applied force was quite linear and repeatable over number of cycles with some hysteresis.

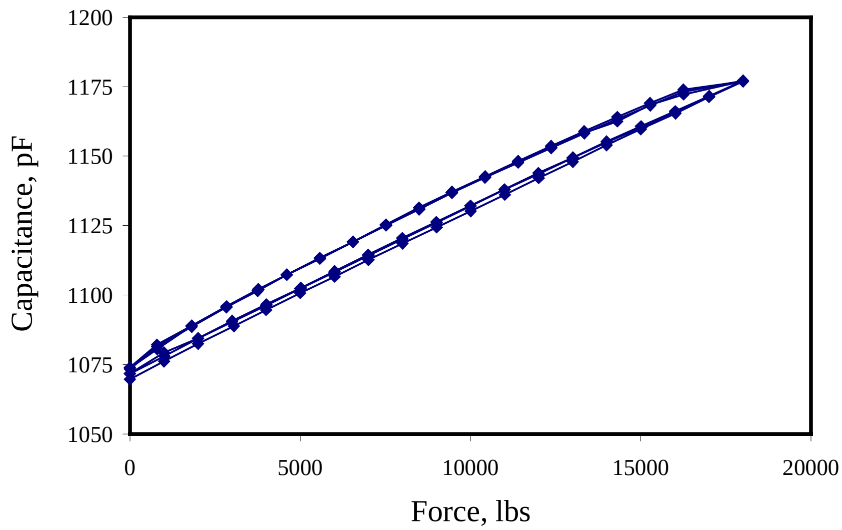


Figure 4: *Calibration plot at 300 K. Note that there are three load-unload cycles shown.*

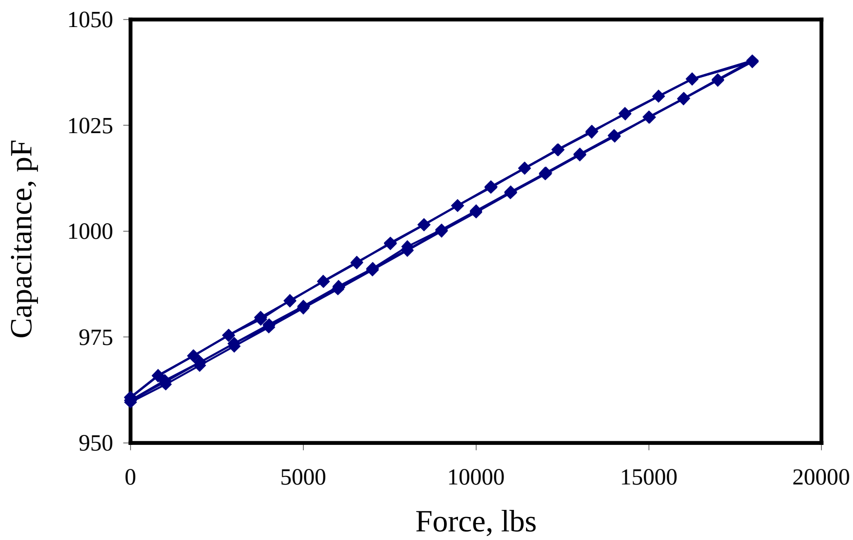


Figure 5: *Calibration plot at 77 K. Note that there are three load-unload cycles shown.*

Four resistive gauges, two each on a spacer were mounted to measure the azimuthal stress near the pole region and near the mid-plane or parting plane region. The instrumentation layout is as shown in the Fig. 6.

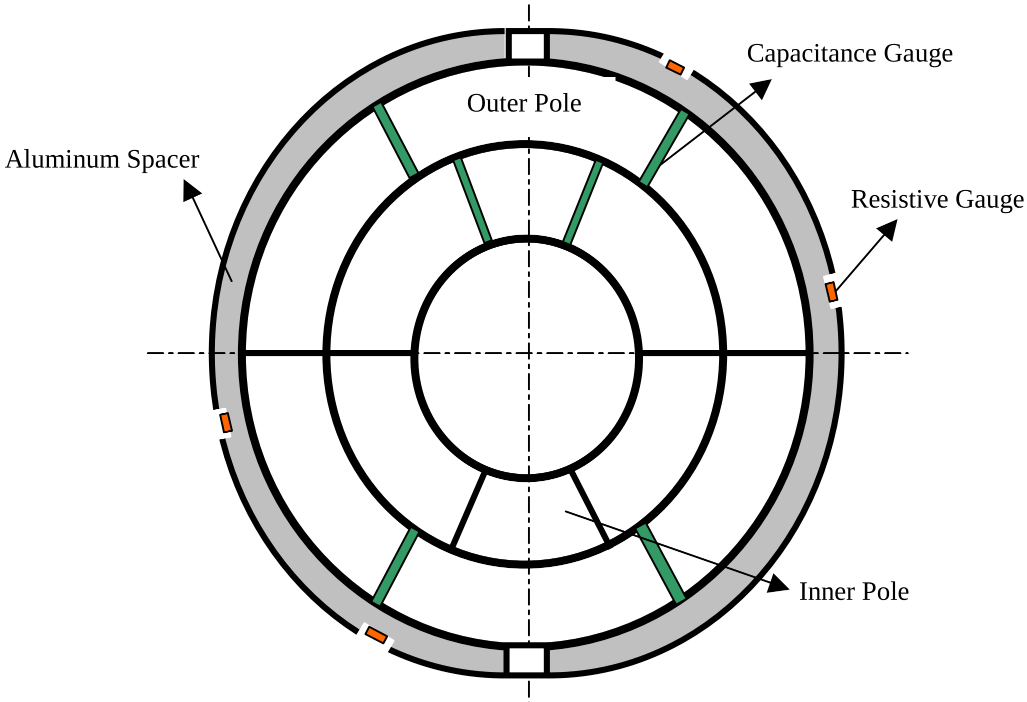


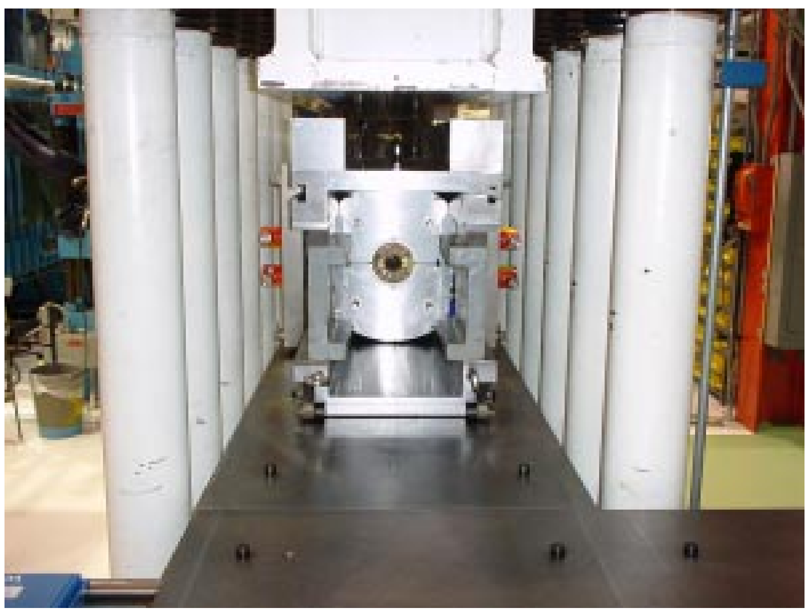
Figure 6: *Schematic of the instrumentation layout in the mechanical model.*

<u>Model Assembly</u>:

The model was first assembled into two iron halves. The new contact tooling was used to compress the yoked assembly and to insert the aluminum clamps. After slight modifications to the tooling the assembly process went without any problems. Fig. 7 shows the model with the contact tooling and inside the press. Supports were placed on either side of the model to reduce any bending in the contact tooling. The top platen was first energized to compress the whole assembly to the right geometry, before the aluminum clamps were inserted into the slots using a separate set of pusher blocks. Once the clamps were inserted the pump pressure was released and the assembly springs back to an equilibrium position.

Due to slightly bigger diameter of the impregnated coil, we could not compress it to the nominal dimension. Thus the vertical gap between the two iron halves after inserting the clamps was measured to be 1.5 mm instead of nominal 0.75 mm. The fact that we were still able to insert the clamps means that we have increased the interference between the clamp and the yoke and should expect higher prestress in the coil than the nominal values. This increased interference was taken into account while doing the analysis.

(a)



(b)

Figure 7: *Magnet assembly process (a) model with contact tooling and (b) model with contact tooling inside the press*.

Test Results and Comparison with the ANSYS Analysis

Data was collected from dial indictors, capacitance gauges and resistive gauges during all stages of magnet assembly to understand the behavior of the magnet.

Coil Rigidity:

Dial indicators were placed on either side between the two platens that push the yoke halves. The pump pressure was increased incrementally (500 Psi steps) and the displacement was noted. This gives us the rigidity of the impregnated coil structure. Figure 8 shows the data. Note that the response was quite linear (the two lines represent measurements taken on either side of the model). Something which is not shown in the plot is that these measurements were quite repeatable. This indicates that we are working in a elastic regime of the coil structure. Also shown in the figure are the ANSYS predictions. The analysis tracks the experimental data reasonably well. However there is difference in slops between the analysis and the experimental data. The analysis predicts a stiffer coil structure than the experiment.

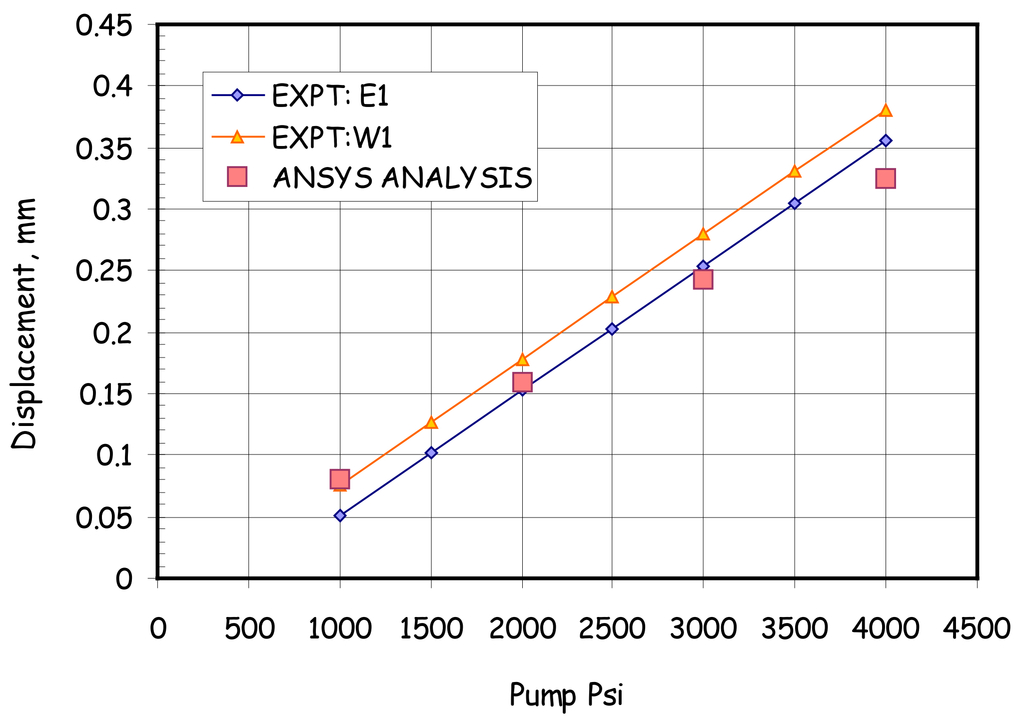


Figure 8: *Displacement Vs Pump Psi. This data provides an estimation for coil rigidity*.

Azimuthal Stress in the Spacer:

Four resistive gauges were mounted, two each on a spacer as shown in the Fig. 6 to measure the azimuthal stress. The model assembly was compressed upto 4000 Pump Psi and then the clamps were inserted into the slots. The pressure applied on clamps was about 2500

Pump Psi. Figs. 9 and 10 shows the azimuthal stress during the various stages of the assembly process at the parting plane and the pole respectively. With the clamp insertion the stress in the spacer increases, however after spring back, the stress decreases by about 20%. Also included the figures are the analysis results. Note that the analysis tracks the experimental data reasonably well expect for the spring back data. ANSYS model predicts more spring back than what the test data shows.

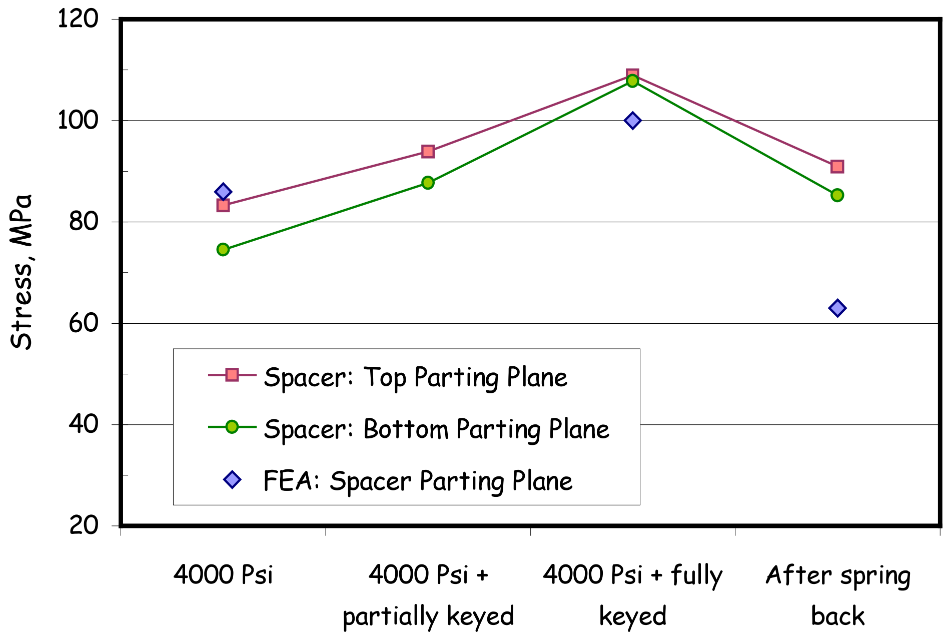


Figure 9: *Azimuthal stress on the top surface of the spacer near parting plane.*

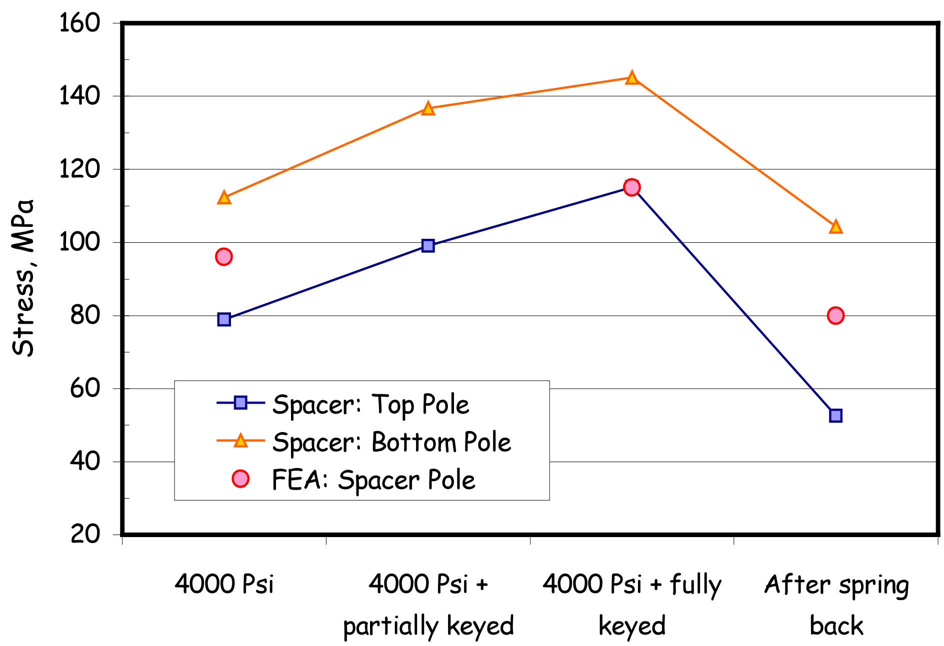


Figure 10: *Azimuthal Stress on the top surface of the spacer near pole region.*

Stress in the Coils:

Capacitance gauges were used to measure the azimuthal stress in the coils in the pole region. Figs 11 and 12 show the stress values in the coil at various stages of the magnet assembly.

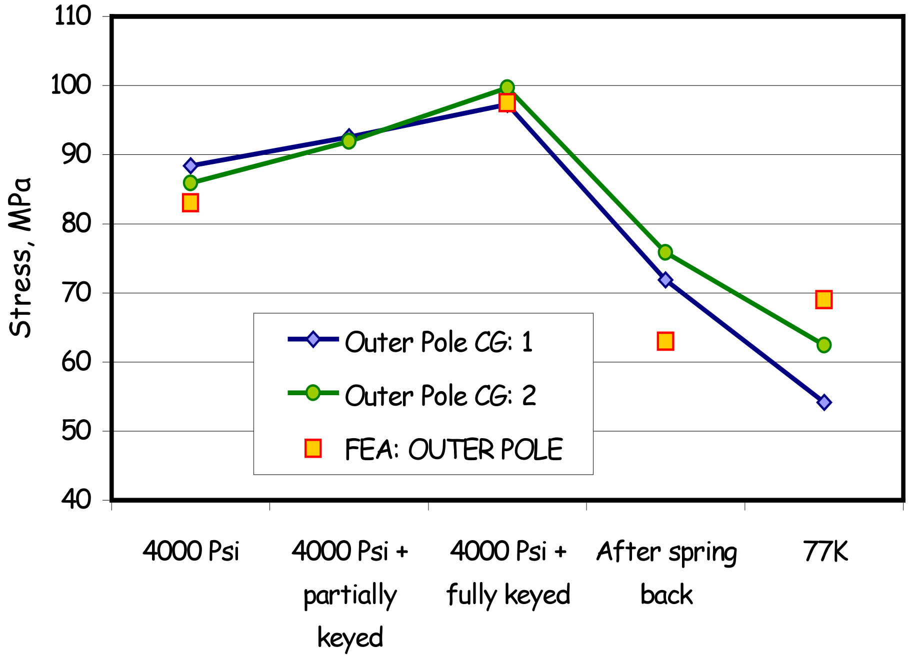


Figure 11: *Azimuthal stress in the outer coil near the pole region.*

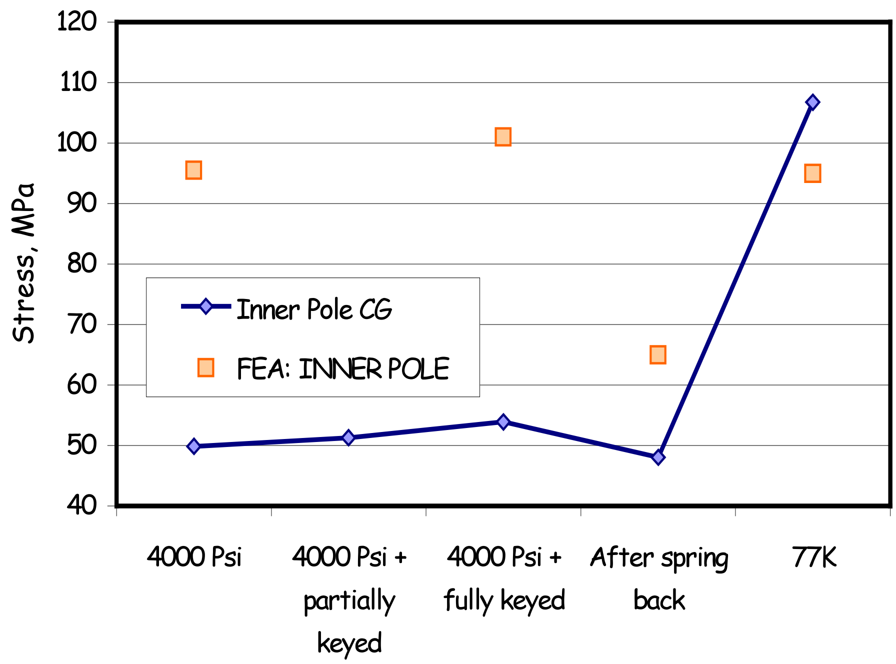


Figure 12: *Azimuthal stress in the inner coil near the pole.*

As expected the stress in the coils increases during the insertion of the clamp and decreases after spring back. The amount of spring back in the coils is about 25% (based on outer layer data). Also shown in the figures are the ANSYS predictions. The analysis agrees reasonably well with the outer layer data. However for the inner layer, the analysis predicts much higher stress than what the gauges read. Its possible that the asymmetry in the inner layer due to the removal of a turn in the mid-plane might have caused this discrepancy.

The above figures also show the data on cool down to 77 K. We obtained this by immersing the whole mechanical model in a liquid nitrogen bath. The stress in the outer layer pole region dropped from about 75 to 60 MPa where as the ANSYS predicts about the same stress levels at room temperature and at 77 K. For the inner layer, analysis predicts a jump in the stress from about 65 to 95 MPa on cool down to 77 K. The experimental data also shows increase in stress from 50 MPa to about 107 MPa.

It should be noted that we should not make final conclusions from the comparisons between the analysis and the experimental data from this model due to asymmetry in the inner coil and the fact that the yoke gap did not compress to the nominal size. However at this point what we could say that the magnet assembly procedure and the contact tooling works, the instrumentation plan seems to be good and finally the analysis is in the right ball park. A second mechanical model is in the pipeline which will be used to accurately test the analysis.